HINDU COLLEGE, GUNTUR

(Re-accredited by NAAC as Grade ‘A’ with CGPA 3.07)

DEPARTMENT OF ELECTRONICS

(UG PROGRAMM)

ADD ON PROGRAMM

(VALUE ADDITION PROGRAMM)

PRINTED CIRCUIT BOARD DESIGN

&

FABRICATION

ADD ON PROGRAMME

ENROLLMENT OF STUDENTS

S.NO	NAME OF THE STUDENT	ROLL.NO	PROGRAMM
1	B. SAI SEKHAR	474	PCB
2	CH.SURYA ROHIL	479	PCB
3	CH.JYOTHIRMAY	480	PCB
4	K.JYOTHI	493	PCB
5	M.SIVA KOTESWARA RAO	498	PCB
6	K.PAVANI	489	PCB
7	T.SUDARSHAN	514	PCB
8	D.MADHU BABU	485	PCB
9	P.NARENDRA	506	PCB
10	M.MALLIKARJUNA ACHARI	495	PCB
11	A.VENU BABU	471	PCB
12	V.SRINIVAS	527	PCB
13	P.SIVA ANJANEYULU	505	PCB
14	S.ANIL KUMAR	517	PCB
15	M.UDAY BABU	501	PCB
16	J.PAVAN KUMAR	487	PCB
17	J.KALYAN	488	PCB
18	Y.RAJA	531	PCB
19	M.PRADEEP	499	PCB
20	NARAYANA REDDY	521	PCB

SYLLABUS FOR PCB DESIGN

&

FABRICATION

Name of Program: PCB Design & Fabrication Duration: 24 Session		
S.No:	**Session**	**Requirement**
1	What is this program about? Importance of PCB in electronics? • What is PCB? • Why PCB? Advantages Of PCB	• Projector • Basic Electronic Lab • Class Room
2	Parameters of Electronics • Voltage • Current • Power • Ohm's law Knowing basic Electronic Components • Resistors • Capacitors • Inductors • Diodes • Transistors	• Projector • Class Room
3	Soldering and it technic • What is Soldering • What is Soldering Iron, Flex and Lead? • DIP & SMD Soldering	• Projector • Basic Electronic Lab • Class Room • PC / Computer lab With internet
4	Designing Tool • What is PCB Designing? • Need of PCB Designing? • Which Software used PCB Designing • Why EasyEDA • Basic of EasyEDA (Tools and Options) • How to Draw Schematic in EasyEDA	• Projector • Basic Electronic Lab • Class Room PC / Computer lab With internet

5	Schematic • If I update the schematic, how do I then update the PCB? • How to rename a Sheet/Page or modify description. • What is the unit of the schematic sheet? How to change schematic unit?	• Projector • Basic Electronic Lab • Class Room PC / Computer lab With internet
	□ For a complex project, I want to split the schematic over several sheets. Does EasyEDA support hierarchy? □ How to change the sheet size and modify the design information. □ How to indicate low electronic level in the Schematic Pin or Netlabel □ I can’t convert schematic to PCB. Why is this?	
6	PCB □ How to change the Units of PCB from mil to mm or inch. □ How to pick and move the components on the PCB canvas quickly. □ How to add test point in schematic or PCB? □ Can I create a PCB without creating schematic? □ How to add more fonts for PCB. □ How to insert an Image/Logo to PCB. □ How to insert a DXF as board outline. How to create non rectangular pcb outline such as round? □ How to add a slot and cut out. □ How to measure dimensions on a PCB. □ How to add more layers. □ How to add solder mask aperture. □ How do I set the dimensions of my PCB in the layout?	• Projector • Basic Electronic Lab • Class Room PC / Computer lab With internet

7	Library and Parts • How to create a schematic symbol library. • How to tag my schematic library symbol. • How to create sub parts for multi-part components. • How to change the footprint for a component. • How to add sub parts to a schematic. • How to create a PCB footprint. • How to change a component's footprint? How to find components/parts/libraries?	• Projector • Basic Electronic Lab • Class Room PC / Computer lab With internet
8	Experiment No:1 Name of Experiment: **LED Resistor Calculation, Serial LED Calculation, Parallel LED Cal.** Circuit Design : **Yes** Virtual Test: **Yes** PCB Design: **Yes**	• Projector • Basic Electronic Lab • Class Room PC / Computer lab With internet
	Fabrication: **No**	
9	Experiment No:2 Name of Experiment: **Transistor Act as a Switch, LED Dimmer using Transistor.** Circuit Design : **Yes** Virtual Test: **Yes** PCB Design: **Yes** Fabrication: **No**	• Projector • Basic Electronic Lab • Class Room PC / Computer lab With internet
10	Experiment No:3 Name of Experiment: **RPS Design and Development.** Circuit Design : **Yes** Virtual Test: **Yes** PCB Design: **Yes** Fabrication: **No**	• Projector • Basic Electronic Lab • Class Room PC / Computer lab With internet

<table>
<tr><td>11</td><td>Fabrication
Different method to make PCB
Toner Transfer Method
Hand Draw Method
Laser Engrave Method
Screen Printing Method
Toner Transfer Method
Design of PCB
Laser printout
PCB material (FR4, FR1, Paper Phenolic)
Cutting
Washing
Print Transfer
Washing
Etching Methods
Fecl3
HCL+H2O2
Ammonium persulfate
Vinegar peroxide</td><td>Projector
Basic Electronic Lab
Class Room
PC / Computer lab With internet
Iron Box</td></tr>
<tr><td>12</td><td>Drill bit select
Drilling (drill machine)
Toner Cleaning
PCB Track testing Tinning
Silk layer Printing
Flex Coating</td><td>Projector
Basic Electronic Lab
Class Room
PC / Computer lab With internet</td></tr>
<tr><td>13</td><td>Components Assembling
Component testing
Pad soldering
Final PCB Test</td><td>Projector
Basic Electronic Lab
Class Room
PC / Computer lab With internet</td></tr>
<tr><td>14</td><td>Experiment No:4
Name of Experiment: Automatic Street Light.
Circuit Design : Yes
Virtual Test: Yes
PCB Design: Yes
Fabrication: Yes</td><td>Projector
Basic Electronic Lab
Class Room
PC / Computer lab With internet</td></tr>
</table>

15	Experiment No:5 Name of Experiment: **Tank Over flow alarm.** Circuit Design : **Yes** Virtual Test: **Yes** PCB Design: **Yes** Fabrication: **Yes**	• Projector • Basic Electronic Lab • Class Room PC / Computer lab With internet
16	Experiment No:6 Name of Experiment: **Rain Alarm.** Circuit Design : **Yes** Virtual Test: **Yes** PCB Design: **Yes** Fabrication: **Yes**	• Projector • Basic Electronic Lab • Class Room PC / Computer lab With internet
17	Experiment No:7 Name of Experiment: **Automatic Street Light.** Circuit Design : **Yes** Virtual Test: **Yes** PCB Design: **Yes** Fabrication: **Yes**	• Projector • Basic Electronic Lab • Class Room PC / Computer lab With internet
18	Experiment No:8 Name of Experiment: **Touch Switch.** Circuit Design : **Yes** Virtual Test: **Yes** PCB Design: **Yes** Fabrication: **Yes**	• Projector • Basic Electronic Lab • Class Room PC / Computer lab With internet
19	Screen Painting (Indusial type Printing Processer) • Film Development • Screen Making • Emulsion Coating • Dry System • Film Exposed on Screen • Screen Washing • Dry • Screen Test	• Projector • Basic Electronic Lab • Class Room PC / Computer lab With internet

20	• PCB Cutting • Select Layer on Screen • PVC Ink (Circuit Layer Painting) • Etching • Select Layer on Screen • Top layer printing • Dry • Select Layer on Screen • Silk layer of PCB Printing • Solder Mask Printing • UV Drying • Thermal Drying	• Projector • Basic Electronic Lab • Class Room PC / Computer lab With internet
21	Components Assembling • Component testing • Pad soldering • Final PCB Test	• Projector • Basic Electronic Lab • Class Room PC / Computer lab With internet
22	Experiment No:9 Name of Experiment: **Water level indicator.** Circuit Design : **Yes** Virtual Test: **Yes** PCB Design: **Yes** Fabrication: **Yes**	• Projector • Basic Electronic Lab • Class Room PC / Computer lab With internet
23	Experiment No:9 Name of Experiment: **Fire Alarm.** Circuit Design : **Yes** Virtual Test: **Yes** PCB Design: **Yes** Fabrication: **Yes**	• Projector • Basic Electronic Lab • Class Room PC / Computer lab With internet
24	Experiment No:10 Name of Experiment: **Audio Amplifier.** Circuit Design : **Yes** Virtual Test: **Yes** PCB Design: **Yes** Fabrication: **Yes**	• Projector • Basic Electronic Lab • Class Room PC / Computer lab With internet

Getting Started With PCB Design

TABLE OF CONTENTS

welcome to *Getting Started in PCB Design*, a high-level introduction to the PCB design process and how PADS®

PCB solutions can improve your design flow.

Perhaps you're new to printed circuit board design, or maybe you're a seasoned professional. It might be that recent changes inyour company have brought you new tasks and responsibilities, including those associated with the design and production of

state-of-the-art electronic products. Or perhaps you might just beinterested in exploring the advanced capabilities now available tohelp you increase your productivity and improve your product's quality.

Regardless of your motivation, in just 60 easy-to-read pages, this guide will show you how to take a design from concept to completion in a format that will prepare you to develop your own designs with confidence and accuracy. Each chapter will start withan overview of basic design concepts, and then proceed with a section on how PADS solutions can be used to streamline the process.

GETTING STARTED IN PCB DESIGN

NEW TO PCB DESIGN?

New to PC

Design?

The Design Process

Any process is easier to learn when there is an underlying structure that holds it all together. Printed circuit board design is no exception; there is a simple flow that keeps the process organized and helps to ensure design success. Once you're familiar with it, you're well on your way to creating consistent designs that can be manufactured and tested easily.

Using the many advanced design capabilities available in the PADS products, you will learn how to set up your working environment, capture your design, verify the integrity of the design, produce the physical layout, and generate all necessary manufacturing outputs. At the same time, you will be establishing a structured and repeatable design process.

As we explore the design process in the upcoming chapters, we'll help you establish the knowledge and confidence you need to accomplish your design and production goals.

The next section outlines a comprehensive process for collecting and organizing your design data. If you’re new to the design process, you’ll find lots of useful information and ideas on how to approach a new design.

Getting Organized

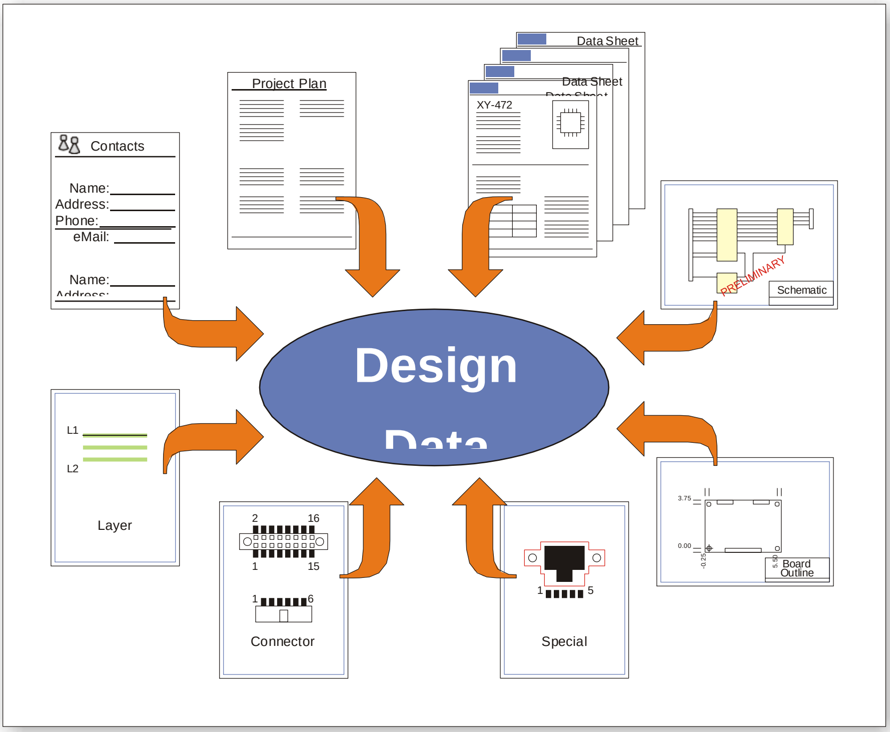
Getting Organized

Before beginning any project, it's a good idea to have a basic plan in place. Regardless of the complexity of your designs, you'll avoid problems and surprises with a little up-front organization. Designing a printed circuit board is a detail-oriented process that brings together information from a variety of sources. As you begin the design process, you will need to have this information available to complete each design task. Rather than interrupting your workflow and concentration each time you need another piece of information, we suggest you develop an organizational system that keeps these important details within reach at all times.

How you store your information is a matter of what works best for you. You may be working in a large organization that requires all information to exist as electronic files that can be shared with other team members. You may be an Engineer working in a research lab where you are the only person who needs access to this information. Perhaps you are a member of a small development team at a start-up company. Collecting and organizing your data is the important step; how you store it is a matter of preference.

This guide offers simple suggestions to help you develop a system for organizing your design and project data. Upon completion of your first few designs, you will notice productivity gains as you develop and improve your own personalized system.

GETTING STARTED IN PCB DESIGN



Electronic vs. Physical Storage

Time and resources typically dictate how you will organize and store your information. Many designers create a 3-ring binder for each project and divide it into sections that contain different types of information. Others create an online directory to store all project-related data. Still others use a combination of techniques. Use the method that works best for you. With high speed networks and Internet connections, much of what you need to complete your design will beavailable in electronic format; however, many companies impose restrictions on Internet access, and not all component information and design data will be available online.

A good starting point might be to create a project binder and keep it near your workstation. Create as much online content as possible, and keep hard copies ofthe most important data in the binder. This way you can have ready access to information during your design sessions and share selected data with others as needed.

Contact Data

At some point in your project you will probably need additional information or answers to project related questions. Because a contact list will save you time when you need it most and help to keep your project moving forward, create a simple form that lists all of thepersons involved with the project and their contact information. You should probably include business addresses, phone numbers, and email addresses.

Depending upon the size of your organization and the scope of your project, this list could include the Project Manager, Design Engineer, Mechanical Engineer, manufacturing personnel, board vendor(s), contract manufacturers, and/or customers.

Project Information

We suggest that you give each design a unique means of identification so that you can locate it easily as it moves through the design process. Individual companies typically develop a system of identification that works with their design, documentation, purchasing, and project management systems. As a minimum, your system should include the project name, board name, board description, board part number (for the unpopulated printed circuit board), assembly number (for the finished PCB assembly), and revision levels. Tryto develop a system that allows for multiple versions andrevisions of the boards. This will enable you to accommodate future design changes as well as alternate assembly versions of your product.

Preliminary Circuit Design

A preliminary design could be a few hand-drawn sketches, a formal schematic drawing, copies of circuit designs from component manufacturers' data sheets, or pages from application notes. If you are not the project Engineer, try to obtain as much detail about the design asfeasible so that you can complete your work with as few interruptions as possible. If your organization has a standard for schematic layout and documentation, we suggest that you review the necessary requirements to make sure you have all of the information you need.

Component Data Sheets

Component data sheets provide information that you will need to complete each of your design tasks successfully. You will extract this information and place it into your design at various stages. In particular, you will need the name of the component manufacturer, part number, schematic or logic symbol, pin numbers, signal names, physical package data, and attribute data, such as voltage, tolerance, and value.

Most component data sheets are available in PDF format from manufacturers' web sites. Many multiple-page data sheets contain information that you will not need for schematic/board design, so examine each sheet carefullyto determine if it contains data important to your design.A good system is to copy the PDF file onto your workstation or file server and then print and store in your binder only the pages that contain design data you need. This will make it easier to find information when you need it. Later, if you find that you need additional information, you can open the data sheets on your system to find the additional data you need.

Board Outline Drawing

To create the physical shape of a board in the design database, you need to know the physical dimensions ofthe board. In addition, you need to know about any cutouts, slots, mounting hole locations and the exact position of any fixed-location components, such as connectors, switches, LEDs and/or panel components; the proposed layer count; and the board thickness.
Typically, this information is provided by a mechanical engineer in the form of a board outline drawing. Ideally, this drawing should be dimensioned from a mounting or tooling hole, and not from a corner of the board.

Layer Stackup

As board designs become more complex and signal integrity concerns increase, many designs incorporate multiple signal and power plane layers. A simple 4-layerdesign may consist of signals on the outer layers, and separate power and ground planes on the inner layers.
More complex designs might contain 10, 12, or 16 or more layers arranged in complex, interleaved arrays to control signal integrity and impedance. Your designs may vary and contain a special layer stackup requirement.

If you are not the Engineer on your project, arrange a meeting early in the design phase to determine the layer stackup requirements. Also, if you have controlled impedance signals, work with your board vendor to establish how they will match their available layer materials with your manufacturing requirements. You will need all of this information to properly create your fabrication drawing(s) later in the process.

Special Footprint Requirements

Before you can transfer your schematic design to a physical board layout, every component in your design will require a footprint. Though many of the componentsyou use in your designs will match with industry footprint standards (such as IPC-7351), other components will be unique, and you will have to construct a custom footprint for them before you can place them into your board layout.

As you gain experience and complete more designs, youwill build and accumulate a library of commonly used parts. Each time you gather the data for a new design, you should check your component footprint requirements against the contents of this library. Any components that are not contained in your library will need to be constructed. Make sure you document the requirements and verify that the component data sheets that you have collected contain all the physical component dimensions that you will need to construct the footprints. Later in this guide, we'll introduce you to the Decal Wizard that you can use to quickly build manyof your required footprints.

Connector Pinouts and Numbering

When you place a connector in a schematic design or a board layout, specific signals need to be connected to particular pins on those connectors. Manufacturers use different methods for numbering pins on similar connectors. Also, mating connectors may use reversed numbering schemes.

10

In order to prevent possible errors in pin numbering, closely examine the manufacturers' data sheets for pin numbering, and then verify the correct numbering assignments, or consult with the Project Engineer. As an additional precaution, verify that thenumbering in your schematic symbols and your PCBdecals are in agreement. Later, when you place your connectors into a board layout, you can use this data to label the pin numbers on the end pins of each connector to aid with pin identification during test/debug cycles or servicing.

Design Rules and Constraints

Design rules enable you to specify the routing requirements for your designs. The manufacturing and design processes typically dictate specific requirements, including minimum trace width, minimum spacing, recommended via sizes, and power trace widths. Your designs may also include requirements for critical signal trace routing widths as well as special rules for individual nets, including maximum length, matched length, controlled impedance, and differential routing rules.

If you are not the Engineer on your project, arrange to meet early in the process to determine the routing requirements for each type of signal used in your design. Each signal type, such as clock lines, power buses, data signal, address lines, and communications signals, can all have different and distinct requirements, so be sure to document all of the rule requirements.

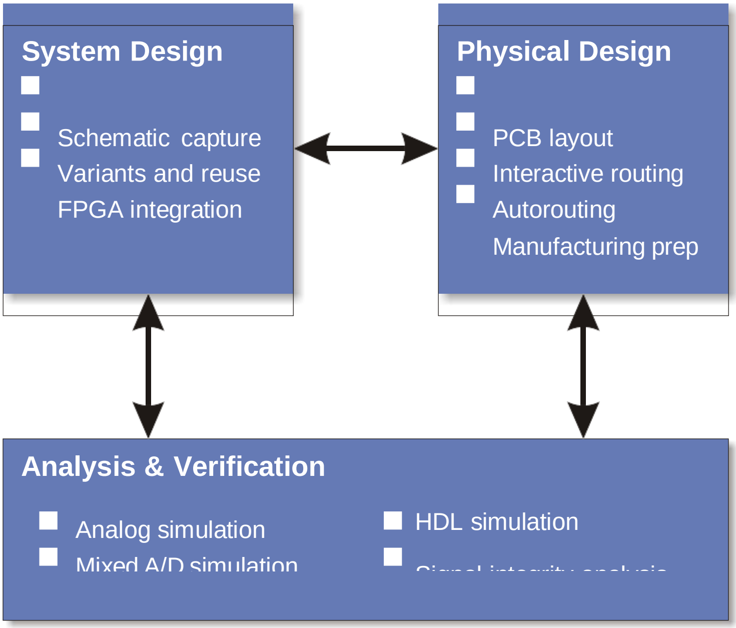
Design Flow

Application Integration

PADS PCB Design Solutions provide a highly integrated environment in which all applications communicate information seamlessly, allowing continuous design synchronization. Changes are passed between the schematic and the PCB layout, allowing instantaneous updates in both forward and backward directions.

PADS

Design Flow



PADS solutions share a common interface design, enabling you to become productive in each program quickly. Data can be seamlessly passed between the design applications as well as simulation programs.

Design Process Overview

Taking a new design from concept to completion requires a designer to pay attention to many details. We all know that designing a printed circuit board is a continuous process of making design decisions and tradeoffs. Throughout the design process, you must weigh a number of conflicting factors and make calculated design choices in order to obtain the best possible design outputs.
Knowing that your design tools have the ability to provide and manage the required content gives you a strong foundation on which to build your design. PADS PCB Design Solutions offer a fully featured front-to-back design flow thatallows you to balance rules and constraints in an intelligent and predictable manner to produce quality designs that can be easily produced.

Though there are hundreds of operations you must perform to produce a final design, we can distill the process down to a few basic steps:

1. Gather the required library parts.
2. Capture the schematic.
3. Generate a netlist.
4. Construct the board outline.
5. Set up the design rules and constraints.
6. Place the components.
7. Route the board.
8. Verify the design.
9. Generate the CAM documents.

Partitioning your workflow to align with these steps will help you establish a structured approach to organizing your design tasks. It will also provide you witha number of checkpoints for reviewing your design data.

The Design Process

Gather the required library parts

↓

Capture the schematic

↓

Generate a netlist

↓

Construct the board outline

↓

Set up the design rules & constraints

↓

Place the components

↓

Route the board

↓

Verify the design

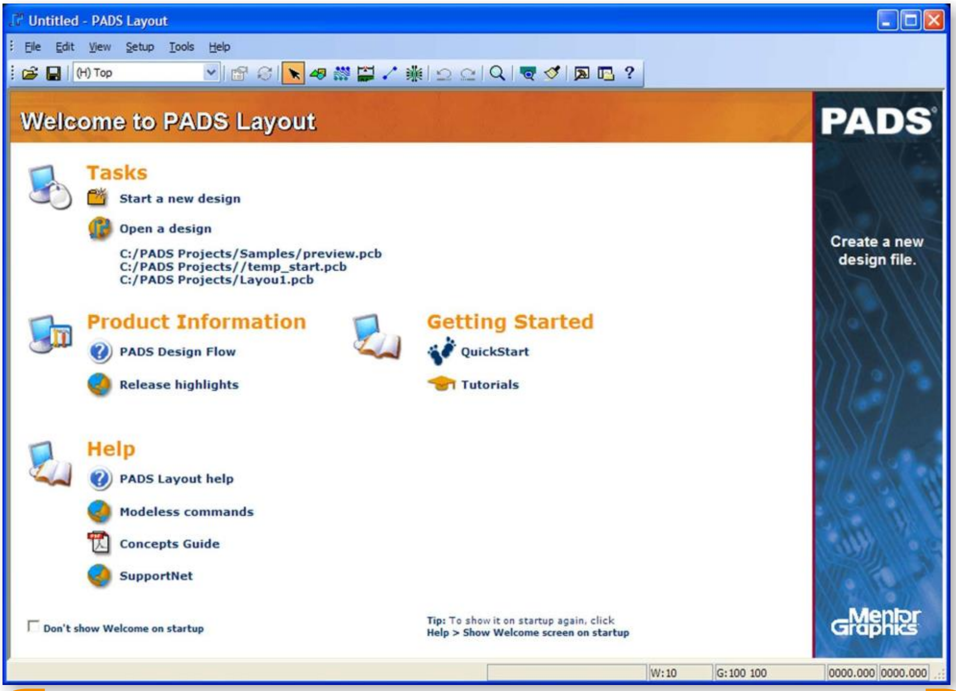
↓

Generate the CAM documents

PADS Environment

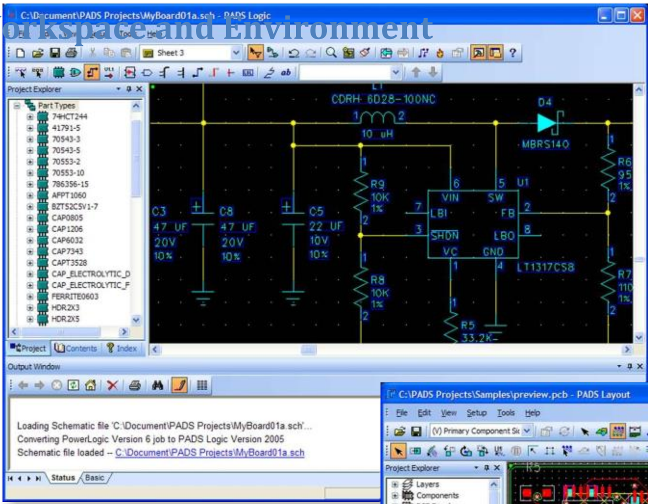
Welcome Screens

The PADS Welcome screens provide information and design navigation ina format that is easy to understand. From this interface you can start a new design, open an existing design, view a product tour, examine a QuickStart, take a tutorial, browse the Help system, review a Concepts Guide, or contact Technical Support.

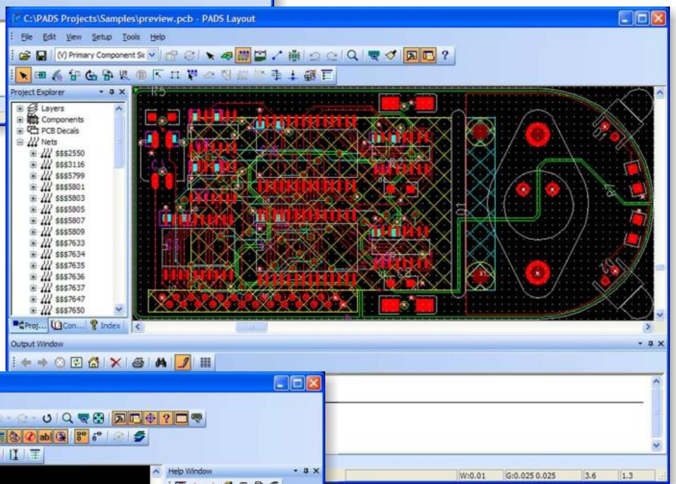


The PADS Welcome screens are your gateway to product information and support. The Help system is your greatest resource for accurate, context-sensitive information about how to accomplish your design related tasks.

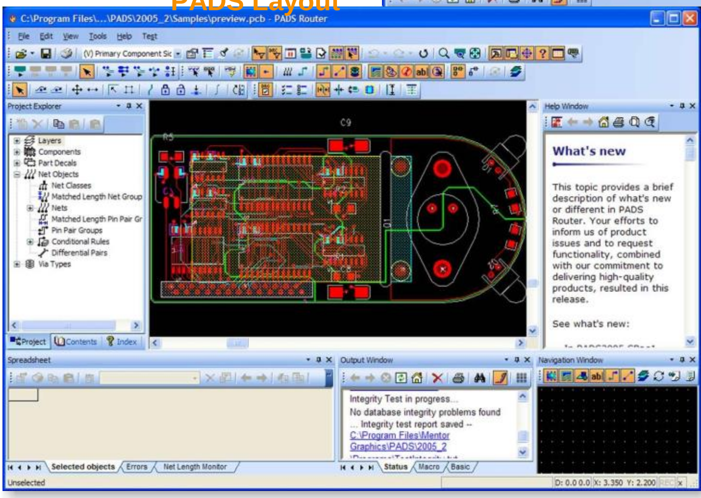
Workspace and Environment



PADS Logic



PADS Layout



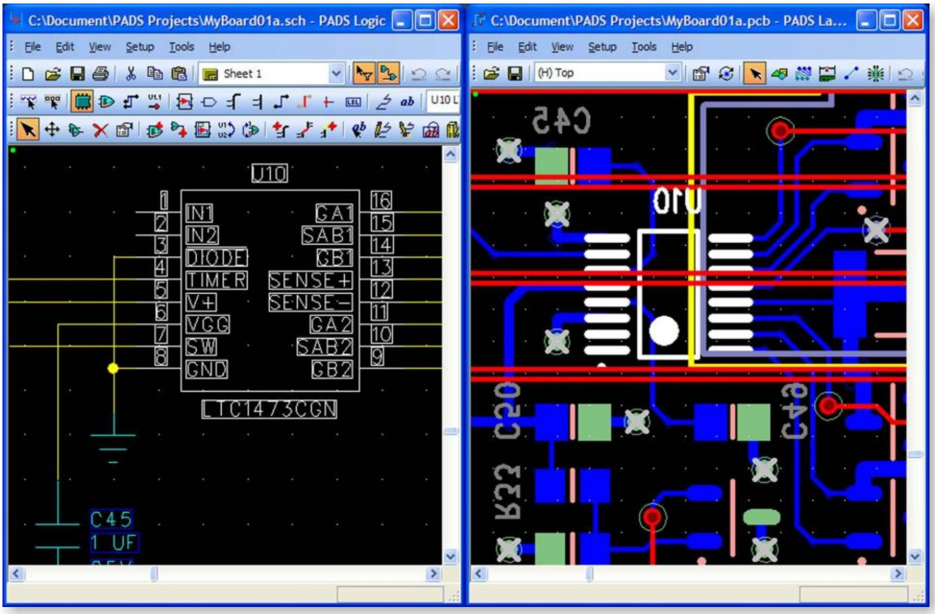
PADS Router

.

Application Links

Once you begin your design, you can easily transfer and share data between the applications. This includes cross-probing from schematic to PCB layout, ECO files, passing data to and from the auto router, and sending data to simulation programs.

Here is an example of multiple linked applications communicating design data. A symbol is selected in the PADS Logic schematic on the left. The decal representing the part is then automatically selected in PADS Layout on the right. This feature enables you to select parts in the schematic for placement in Layout,or to select nets in the schematic to review their routing paths in Layout.



Cross-probing between the Schematic and Board Layout

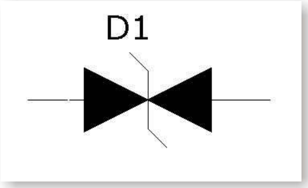
File Types and Locations

All information associated with a design is stored in the PADS database files.This means that each of the design file types (.sch for schematic and .pcb for layout) contains all of the necessary information to open and edit the files.
Because no external libraries or design elements need to be included, it is easy to share or archive design databases. All design files can be output asASCII files forediting and transfer of data to other applications.

Library Parts

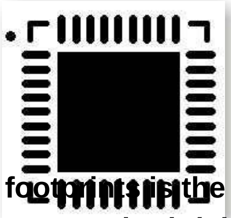
Library Parts

Every design is based on the interconnection of a number of separate components into a functional product. A schematic diagram uses graphical symbolic representations of the electrical or logical functions of the components.



Organizations such as ANSI and IEEE oversee various standards for the proper construction and presentation of these symbols. Typically, you will find graphical representations of the parts on the component data sheets that you collect during the organizational stage of your project. You will use these as models to construct your schematic symbols (CAE decals).

In addition to the graphical symbols used in the schematic, each component must have a physical representation of the part (a PCB decal) to place onto the printed circuit board layout.

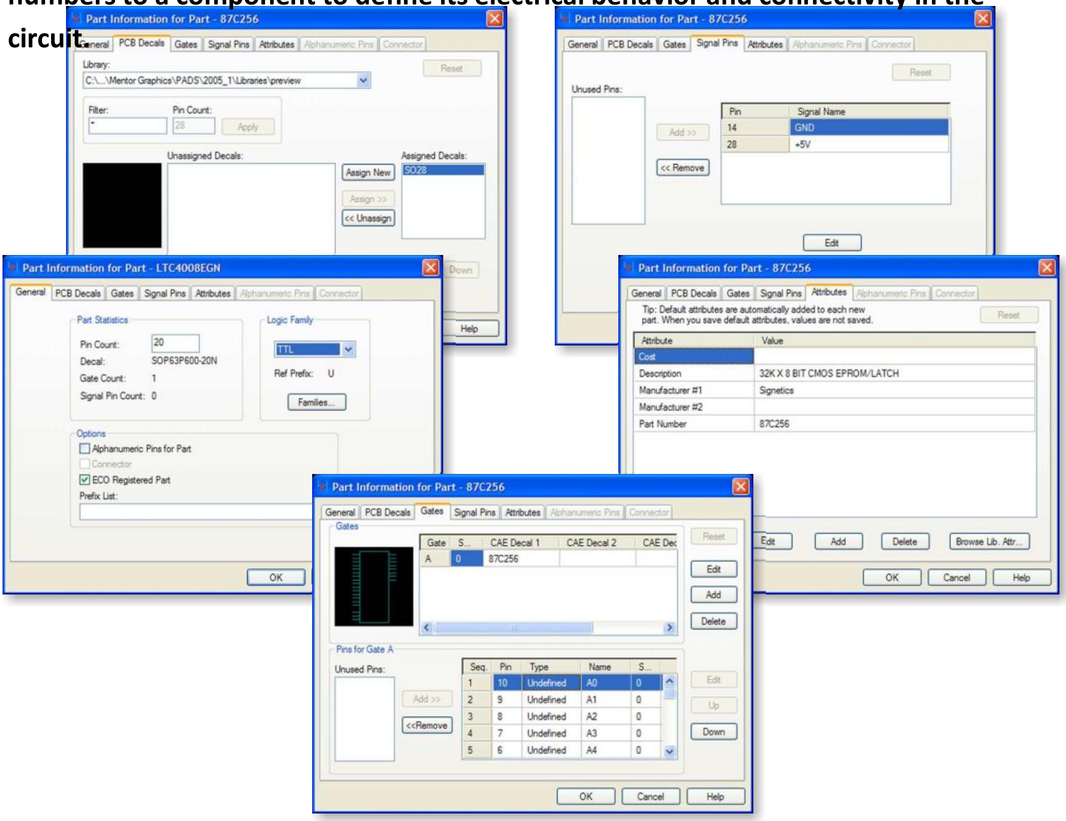


The most notable standard for the construction of these footprints is the IPC-7351 standard, which recently replaced the IPC-SM-782 standard. (The reference section at the end of this guide provides more information on these standards.) The data sheets that you collect also contain the physical dimensions of the components. In addition, some may contain recommended patterns for the decals. Use the physical dimensions along with construction guidelines from your preferred standard to build the decals for your components.

The PADS libraries include thousands of common electronics components. You can use the Library Manager to browse this comprehensive collection of components. You can also use the Symbol and Decal Wizards to build new parts quickly.

Electrical Properties

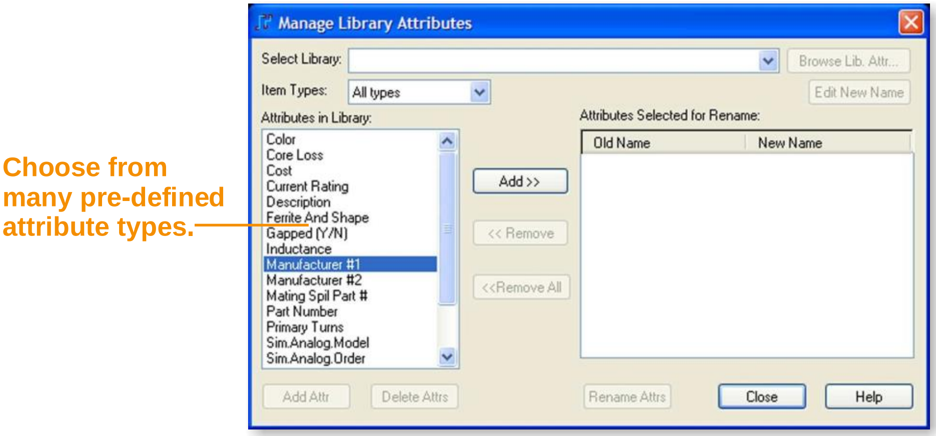
Electrical properties allow you to assign decals, gates, signal names, and pin numbers to a component to define its electrical behavior and connectivity in the circuit.



Electrical Properties Assignment Tabs

The Attribute Manager

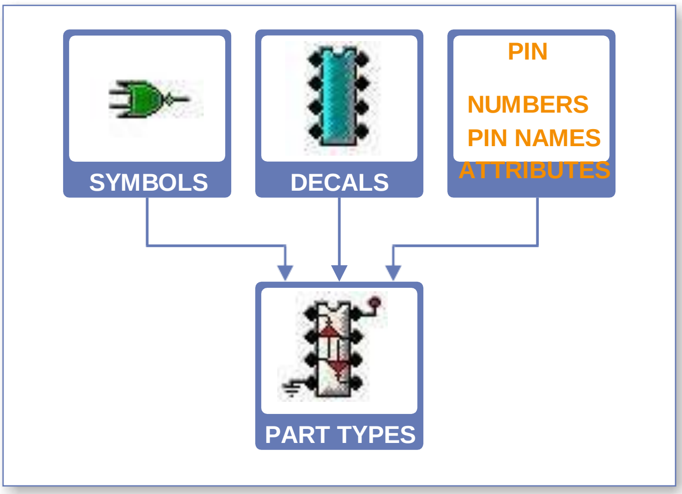
You can also assign attributes such as manufacturer, part number, description, component height, value, tolerance, and cost to each component. PADS is pre-configured with many attribute types to choose from, and you can easily add



The Attribute Manager

If your design contains parts from a variety of sources, each with differing attribute names, the Attribute Manager provides you with the capability to achieveconsistency and structure by combining and renaming attribute names and values across your entire design.

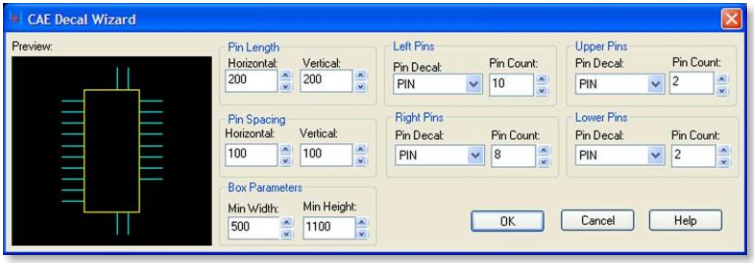
To simplify the process of organizing all of this information, PADS allows you tolink symbols, decals, electrical properties, and attribute data together into a common element referred to as a Part Type.



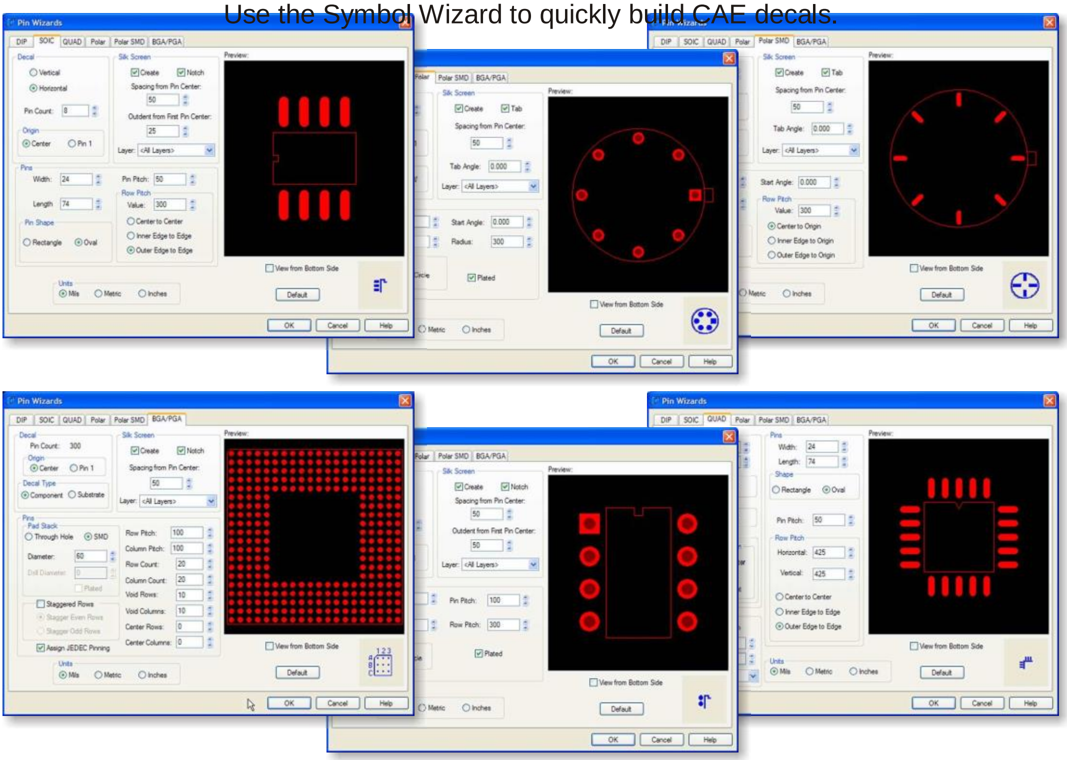
Library Objects Associations

Part Types and Wizards

Each PADS library contains a collection of related Part Types, providing you with thousands of parts to choose from when creating your designs. If you can't locate the parts you need, you can use a number of tools, including the Symbol Editor, the Decal Editor, and the Part Editor, to create new symbols, decals, and part types. In addition, using the Symbol and Decal Wizards enables you to quickly build complex schematic symbols and PCB decals in minutes.

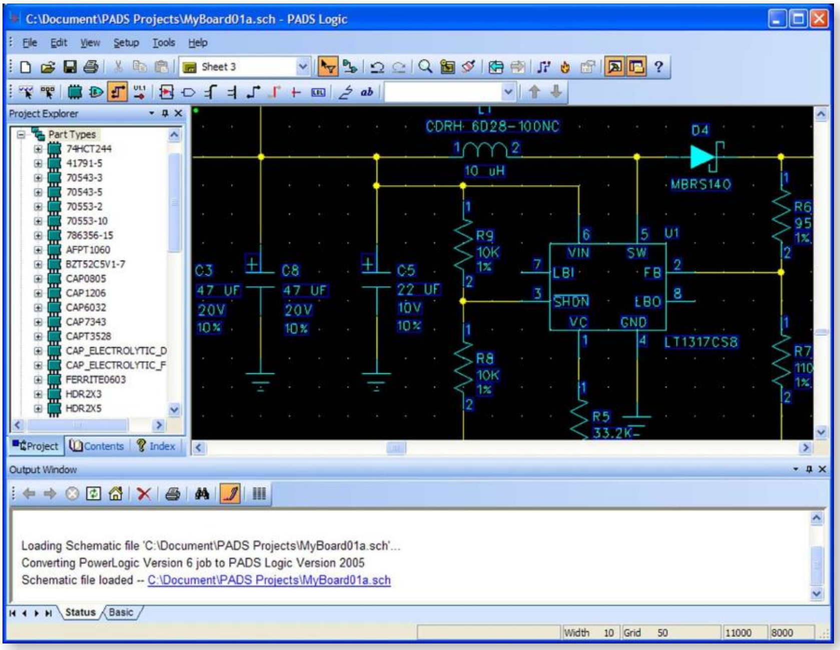


Use the Symbol Wizard to quickly build CAE decals.



Schematic

The Schematic



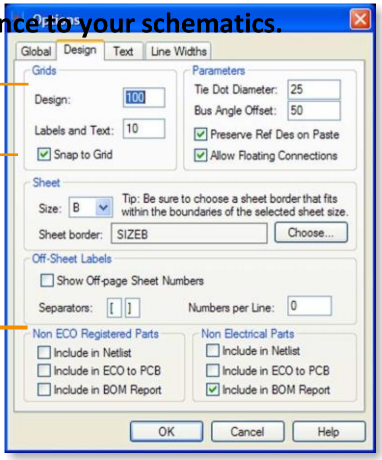
A typical schematic viewed in PADS Logic.

The schematic is a graphical representation of your design that contains the connectivity, part attribute information, and design rules. From this design you can extract information in a variety of formats, such as a netlist of the connectivity, a Bill of Materials, data to send to simulators, and prints or plots of the design. Because this is an intelligent design database, it contains not just a pictorial representation of your design, but the actual interconnection of the components, as well as the component attributes such as part numbers, values, manufacturers, and tolerances.

The PADS Logic Schematic Editor allows you to capture your designs, including support for hierarchical blocks, quickly and efficiently . Use the PDF output option to produce intelligent PDF design files to share with engineers and vendors.

Creating a Schematic Design

You create the schematic in PADS Logic by selecting a sheet size and then placing the desired component symbols on the sheet. Support for TrueType® fonts allows access to a broad selection of Engineering and Scientific notation characters as wellas provides a professional typeset appearance to your schematics.



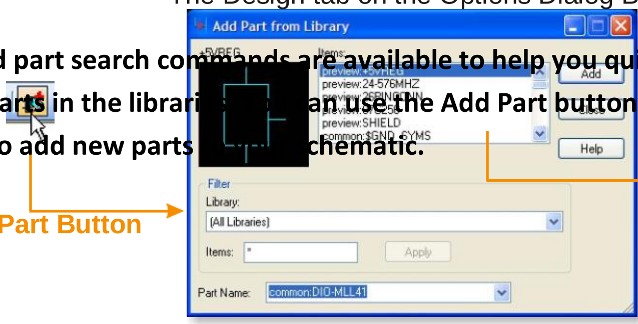
Set the Design Grid

Set grid snap

Set the sheet size

The Design tab on the Options Dialog Box

Advanced part search commands are available to help you quickly locate the desired parts in the librari[illegible]an use the Add Part button on the Design Toolbar to add new parts [illegible]chematic.



Add Part Button

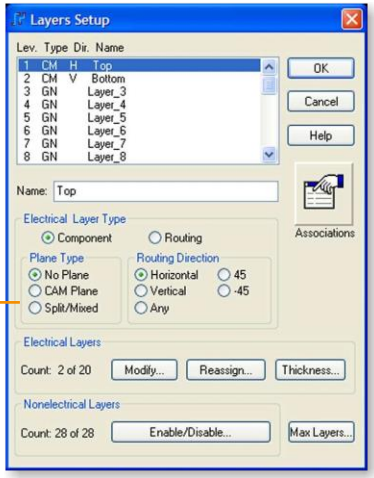
Selection List

Special symbols are available to indicate power and ground connections. Supportfor off-page symbols includes notation of signal names and sheet numbers for easy signal identification across multi page schematics.

Layer Setup

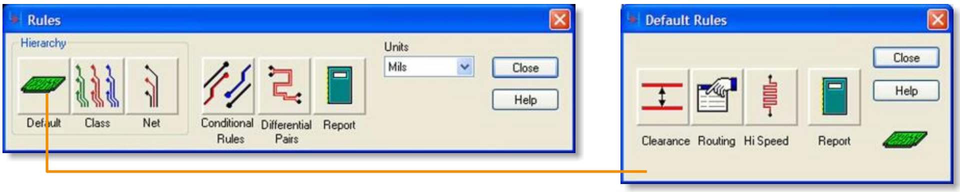
You can set up your preferred layer stackup during the schematic design session ifyou want, or wait until you transfer the design to the Layout environment. PADS offers the flexibility to allow the Engineer or the Designer the ability to enter or modify these settings at any stage of the design development.

Use the Layers Setup to define your layer stackup, including power and ground planes.



Setting up Your Layer Structure

You can also specify design rules to indicate clearances, routing widths, preferred routing layers, and numerous other rules that will be observed by the interactive and automatic routers. In addition, you can set up design rules in Logic, Layout orRouter so you can add or modify your rules at any convenient point in the design process. Using the ECO process, rules can be seamlessly passed between environments.



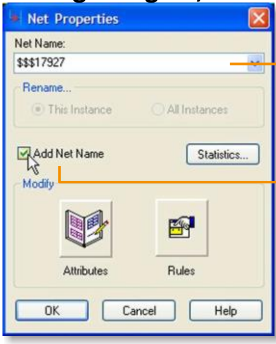
Selecting Design Rules Categories

Naming Nets

You can assign names to nets and buses for identification. After wiring you can edit or change net names. Additionally, you can add or edit component attributesto define component values or part numbers. You can easily modify attributes inLogic or Layout to update your design values as your design changes. Again, these updates are accurately managed by the ECO process.

Attribute

Value Field



Name Field

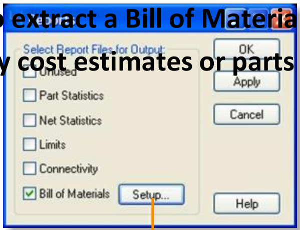
Add Net Name Check Box

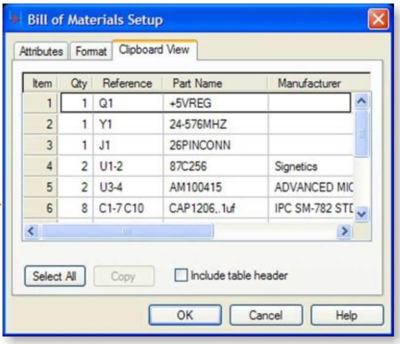
Use Part Attributes to add or change part attributes.

Use Net Properties to name a net.

Reports

Before you begin the physical layout of your design, you can use the Report function to extract a Bill of Materials and generate a component listing for preliminary cost estimates or parts procurement.

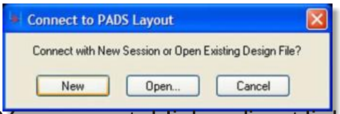




Netlist Creation

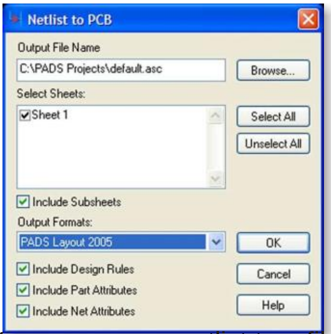
Netlists

Once you have completed and reviewed your schematic design, you can generate a netlist to transfer your design to PADS Layout and create your printed circuit board design. The netlist contains all of the connectivity and component data to represent your design on the PC board.



You can establish a direct link between applications.

You can set up a direct link between PADS Logic and PADS Layout to transfer the data, or you can save the netlist as a file and import it at a later time. If you have both applications on your workstation, you may want to use the link to transfer the netlist from the schematic to a PC board layout in one simple step.



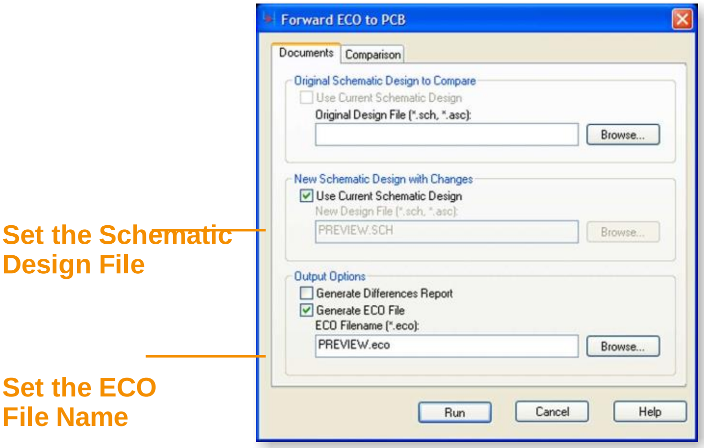
You can save a netlist to a file for transfer to another system.

If the board is being created at a different location or by another person on a different workstation, you can output the netlist to a file and forward the file to the PCB designer.

Engineering Change Orders (ECOs)

The design database is bidirectional; information can be updated in either direction between the schematic and the board layout with Engineering Change Orders. These transfers can occur through a direct link, or you can save ECOs asfiles and import them later into applications when requested. Netlist comparisonutilities also allow you to compare different netlists to identify updates and changes.

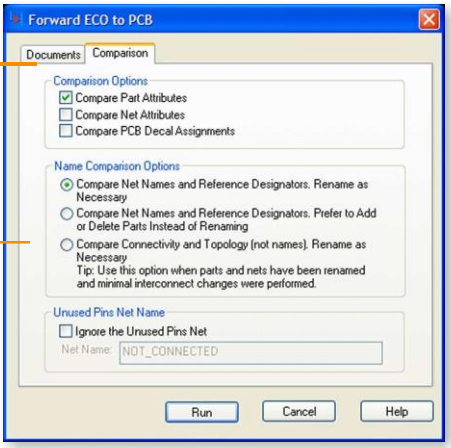
As your design progresses, you will probably make changes along the way. Part values, package sizes, interconnect modifications, and part substitutions all occur frequently in a typical design cycle. Maintaining and keeping this data synchronized between the schematic and the board layout is an important step in the design process. PADS provides complete forward and backward annotation capabilities allowing you to update your board design with schematic changes aswell as send changes from the board design back to the schematic.



Send an ECO to PADS Layout

For example, if you resequence the order of your reference designators on the board, these changes can be sent back to the schematic so that you can correctly identify the parts on your schematic sheets with the updated references. You canuse this same process to update attribute values, part additions/deletions and wiring changes.

You can also compare the contents of two different design files to determine any changes that have beenmade to either of the files. Use the Comparison settings to set up the comparison rules that you wantto apply.



Determine the differences
between two design files.

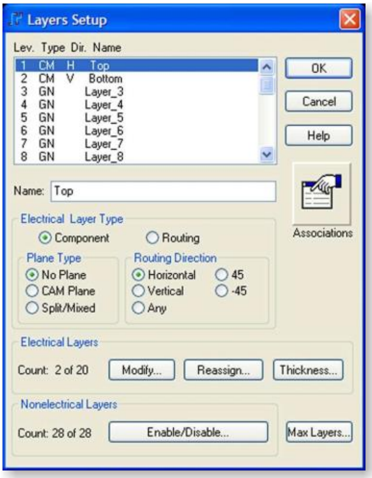
At some point in the life cycle of your product, you may be required to identify specific design revisions.You may have multiple schematic and/or board design files for different revisions of your design andneed to identify the contents of each file. The File Comparison utility allows you to quickly and conveniently identify similarities or differences between files so that you can determine the correct course of action. This valuable resource enables you to maintain proper synchronization of your design files.

PCB Design Setup

PCB Design Setup

PADS Layout provides a number of flexible tools for constructing the board outline. Full support for slots and cutouts lets you construct complexboard shapes. You can also import your board outline from mechanical design packages, such as AutoCAD®, using the DXF Import utility.

The Layer Definition command enables you to specify arrangements of layer stackups that are extensive enough to satisfy your most demanding high-speed design requirements.



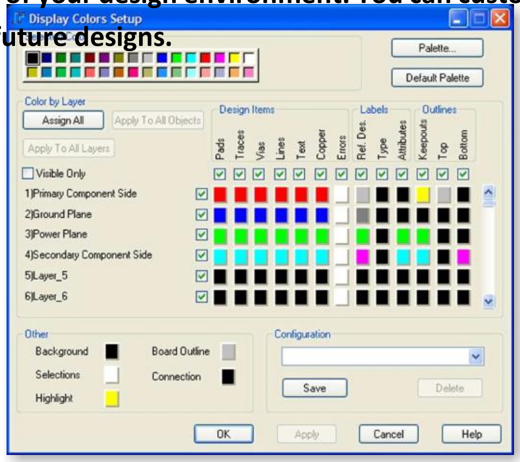
Use the Layers Setup to specify layer types and preferred routing direction.

The layers in your design can be set up for specific purposes, including component mounting, routing, and power/ground planes. You can also specify preferred routing directions for each layer. This allows PADS Router to predictably place traces on each layer, evenly distributing signals while reducing crosstalk.

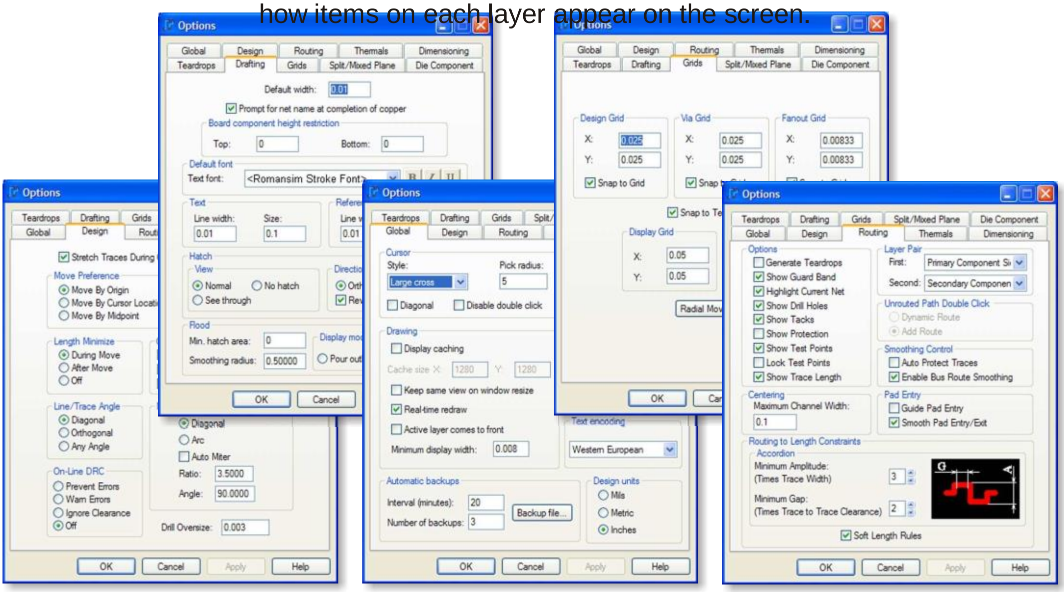
With support for up to 250 layers, PADS Layout can accommodate the requirements of even your most demanding designs. Split-mixed power and ground planes allow you to develop complex power distribution systems to support all of your high-speed designs.

Options

The PADS design environment offers many design options, allowing you complete control over the setup of your design environment. You can customizesettings and save them for use on future designs.



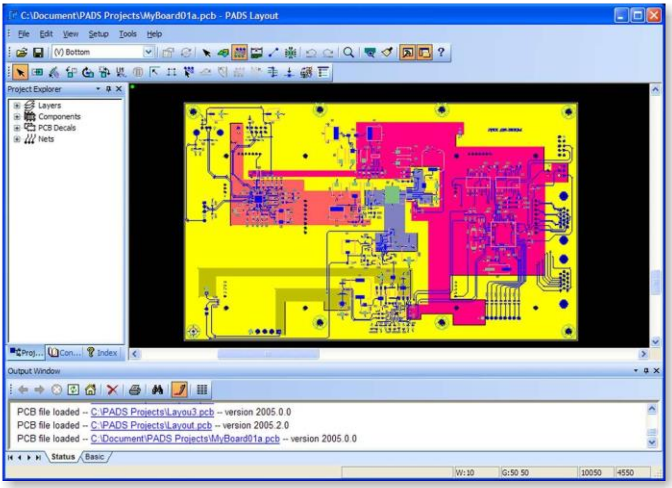
Display Colors Setup allows you to control how items on each layer appear on the screen.



Use the various Options tabs to set up your design environment and to specify your individual routing and appearance behaviors.

Split-Mixed Planes

Split-mixed plane capabilities allow for multi- voltage power distribution schemes, the arrangements of which you can re-specify as your design needs change. Interactive editing of plane splits allows you to adjust plane divisions at any timeduring the design process.



An Example Design Showing a Split Plane
with Five Separate Voltage Areas

Dimensioning

Once you've finished specifying the board shape, an array of autodimensioning tools is available to dimension your board for your fabrication and assembly documentation.

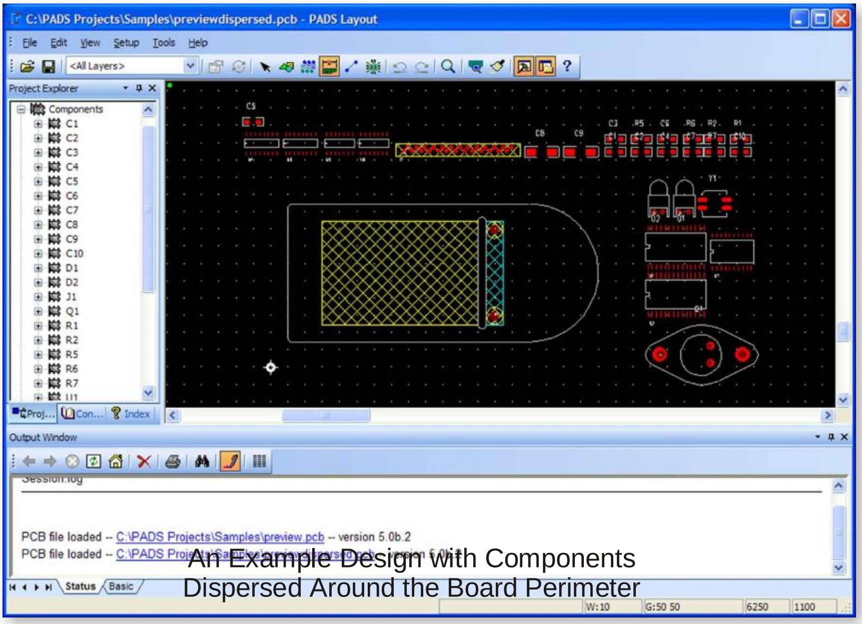
The Autodimensioning Toolbar

Placement

Component Placement

Estimates suggest that most designers spend more than half of their design time on component placement. This is understandable, since a well placed board is much easier to route. Because we are familiar with the demands placed on designers during placement operations, PADS offers multiple methods for achieving high quality component placements.

After you have your board outline in place and have imported your netlist, all of the components appear in a cluster at the board origin. You can use the Disperse Components command to spread all of the components around the outside of the board perimeter in preparation for placement.

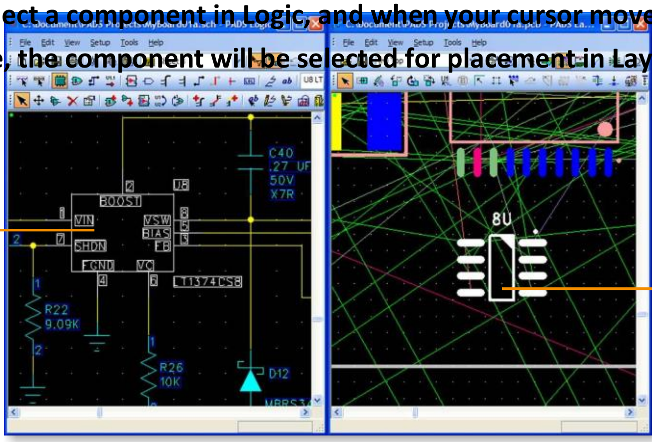


An Example Design with Components Dispersed Around the Board Perimeter

PADS allows multiple methods for component placement including support for cross-probing between the schematic and the board layout. This feature allows you to select and place individual components or groups of components.

Placement with Cross-probing

You can place components in Layout or Router; where you are in the developmentof your design typically dictates your best choice. In either environment, you can establish a direct link with PADS Logic and cross-probe between your schematic and layout. Simply select a component in Logic, and when your cursor moves intothe layout workspace, the component will be selected for placement in Layout.

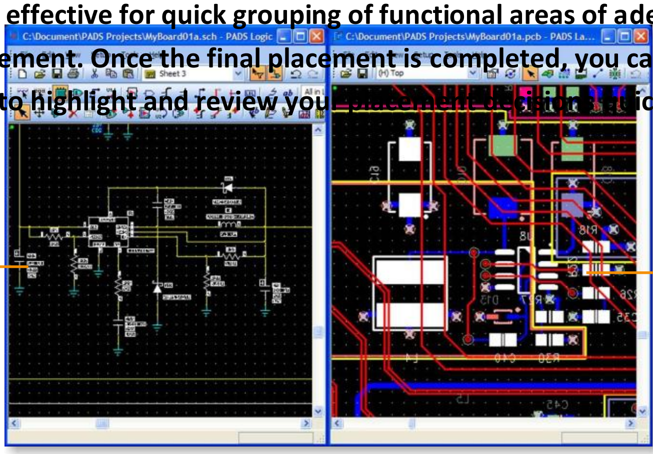


Select a part in the schematic

Part is selected for placement in the layout

Cross-probing with a Single Selection

You can also select an entire group of components in the schematic, such as an entire power supply section, and all of the parts will be selected when you move into Layout. This capability is effective for quick grouping of functional areas of adesign for preliminary placement. Once the final placement is completed, you canuse the cross-probing feature to highlight and review your placement decisions quickly.

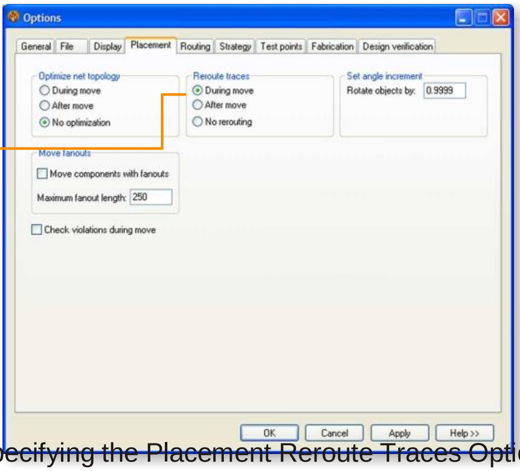


Select a group of parts in the schematic

The group of components is selected in the layout for review

You can also manually place the components one ata time, as well as in groups, using the placement tools within each application. For most designs, placement in Layout is efficient and easy. Typically,you will have an empty board on which you will build up the component placement architecture as you progress.

At times you will need to add or replace componentsin an existing design that includes routed traces. In this situation, the placement tools in PADS Router are your best choice. You can add new components or move existing components while preserving the routed traces, actually pushing and shoving traces and components out of your way as required.

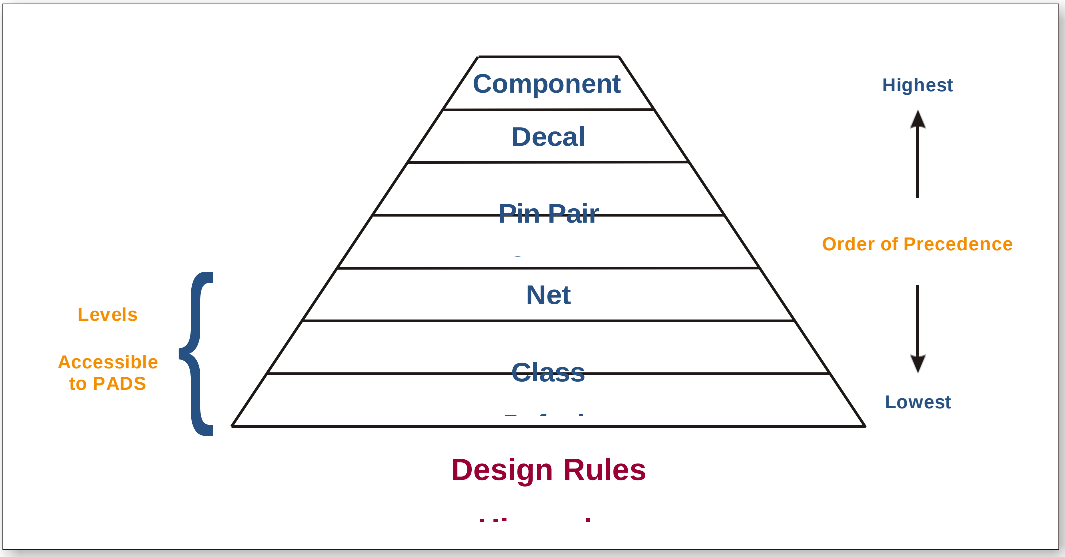


Specifying the Placement Reroute Traces Option

Design Rules

Design Rules and Constraints

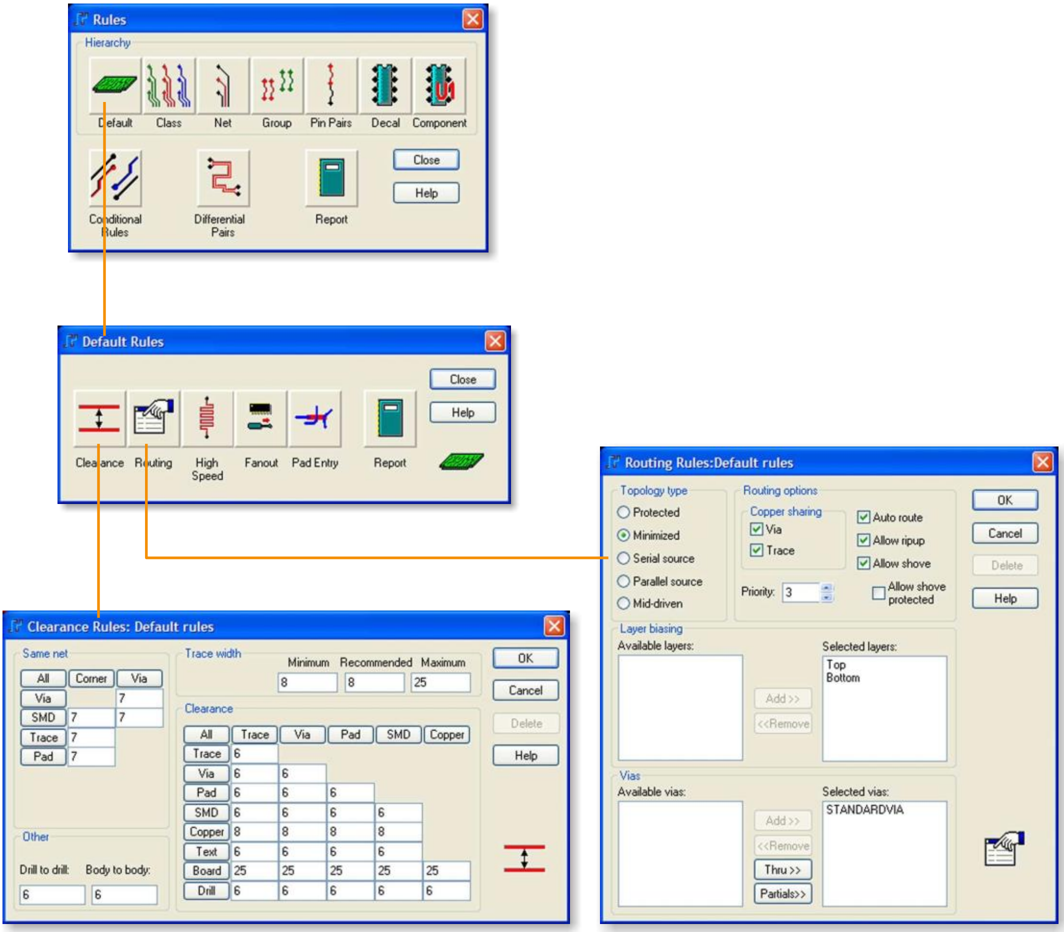
Rules and constraints are what maintain the integrity of your design. With the increasing component and routing densities of today's high-speed designs, you must manage a large number of rules and design constraints. With PADS software, you can specify separate rules for clearance and trace width for individual pin pairs, nets, groups of nets, classes, components, or the entire design if you choose. You have the flexibility to control the length of critical nets or specify matched lengths forgroups of nets.



PADS PCB Design Solutions offer the capability to configure a wide selection of design rules during the design process. This allows the flexibility for the Engineer or the Designer to configure rules during capture, layout, or routing.

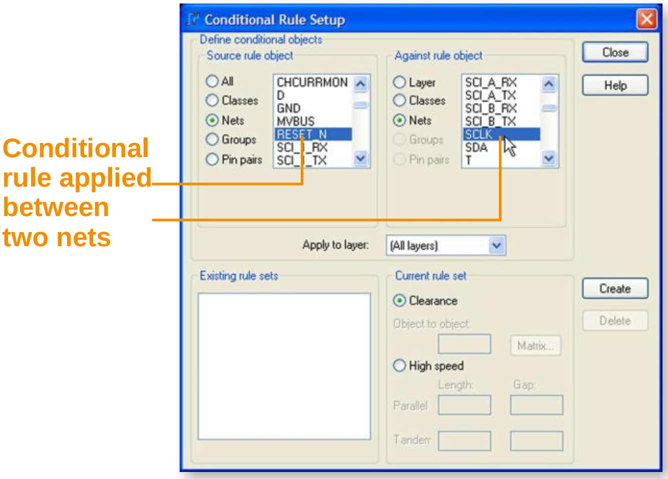
Default Design Rules Setup

You can set up default clearance and routing rules for your design and then build a complex hierarchy of rules as you develop your routing strategy. For example, you might set wider clearances for critical nets such as clocks, or increase the trace widths for your power nets. You can specify rules for classes of nets to forcecertain rules for your address lines while you have different rules in effect for your data lines. These rules are applied during your interactive and autorouting sessions to assure that your design is routed with accuracy and meets your high- speed design constraints.



Net and Conditional Rules

As your design complexity increases, so can your rule assignments. In addition to the default rules, youcan specify rules for individual nets or groups of nets. This gives you the flexibility to define separate rules for specific nets or net groups such as clocks ordata and address buses. These rules have a higher precedence than the default rules, so assigning net rules overrides the default settings.



Conditional Rules Between Nets

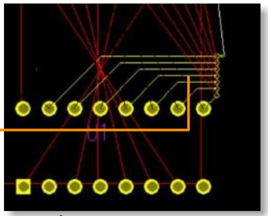
Conditional rules allow you to set up specific relationships between different design objects. Conditional rules can be applied to individual nets, groups of nets, classes, pin pairs, and even to individual components. You could specify that a particular class of nets must observe a clearance of 8mils from each other, but must maintain a clearance of 25 mils from all other signals. This gives you the ability to set up exacting control over the routing of your entire design.

Routing

Board Routing

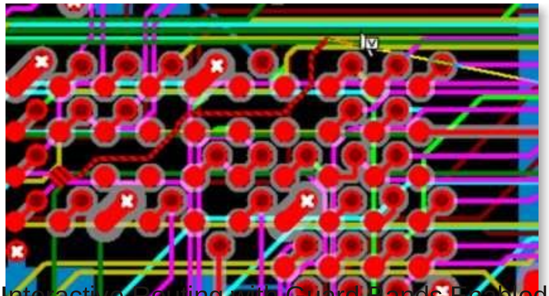
Whether you choose to route your board interactively, autoroute, or use a combination of both techniques, the PADS PCB Design Solutions offer you a number of different routing options. Interactive routing in PADS Layout gives you complete control over routing paths, layers, and via choices. You can select a net to route and then simply choose the desired routing path, clicking to establish each corner as you route toward your destination. You can route individual nets, or use the Bus Router to route entire buses in one operation.

The Bus Router allows simultaneous routing of entire buses.



The Bus Router

In addition to the basic routing capabilities of PADS Layout, PADS Router offers advanced interactive routing features such as trace plowing, push and shove, and high-speed routing capabilities including differential pairs routing and length tuning with accordions. You can route individual nets, all nets connected to a particular component, groups of nets, or even the entire board.



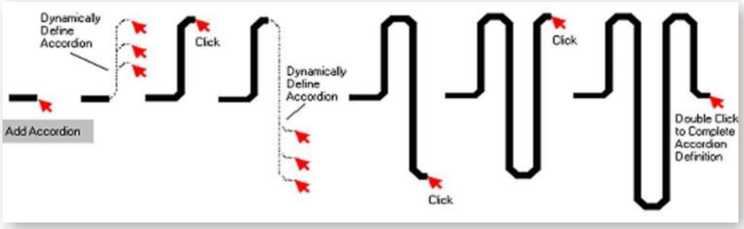
Interactive Routing with Guard Bands Enabled

PADS Router offers multiple interactive and autorouting capabilities, allowing you complete flexibility in the routing of your design. You can easily partition your routing tasks and apply a variety of routing methods to route your board to completion.

High-Speed Routing with PADS Router

PADS Layout offers a selection of interactive routing methods that will allow youto accomplish many common routing tasks. However, if your designs contain high-speed signals, you will want to explore the advanced routing functionality available within PADS Router.

PADS Router offers advanced capabilities for routing high-speed traces. You can control trace length for individual nets or groups of nets. Full support for minimum/maximum matched length groups is available. Length rules that you specify will be observed by the interactive router as well as the autorouter. If you prefer to control the exact positioning of your matched length traces, you can route them interactively and monitor your progress with the Trace Length Monitor. PADS Router lets you add length to traces by interactively creating accordions.

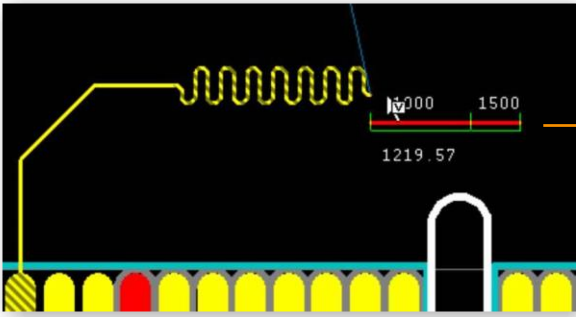


Powerful and Intuitive Accordion Construction Methods

Length tuning with accordions is accomplished with a simple click-and-drag interface that also provides a graphical Trace Length Monitor right at your cursor.There are many options available to control the amplitude and clearances betweenloop segments, but you can use the default settings to create accurate symmetrical accordions with a few mouse clicks.

PADS Router also allows you to specify matched lengths for groups of nets. Specify the length and tolerance for the group, and PADS Router will maintainequal lengths on all of the specified nets. You can also specify min/max lengthswhen you have a wider window of acceptability in your design.

Trace Length Monitor



Visual indication of the length of routed and unrouted trace segments

Routing High-Speed Traces with the Trace Length Monitor

When routing your length-controlled nets, the Trace Length Monitor gives you a visual indication of the length of routed and unrouted trace segments. This helps you to quickly determine when you have achieved the desired trace length.

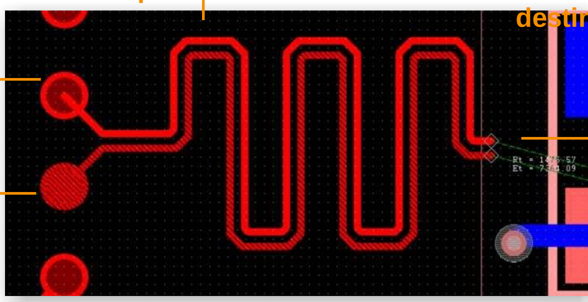
Differential Pair Accordions

If you have differential pairs in your design, you'll appreciate the speed and accuracy you can attain by routing pairs together and even adding differential pairs accordions as you route.

Both signals are simultaneously gathered at the proper gap

Trace gap is maintained throughout the routing process

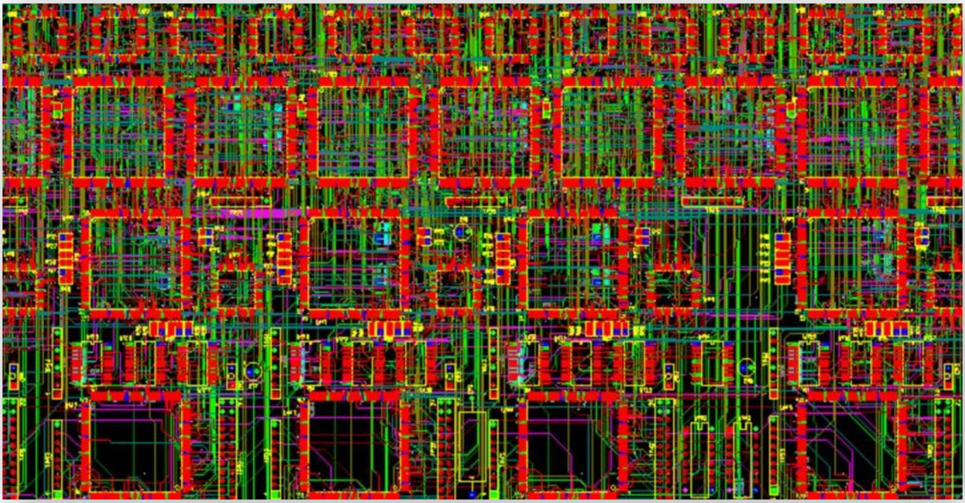
Traces are routed together as a pair from source to the destination



Length Tuning with Differential Pairs Accordions

Autorouting Complex Designs

To meet your autorouting requirements, PADS Router is available to quickly and efficiently route your design. You need only to define your routing rules, select a strategy, and start PADS Router. Regardless of your design complexity, PADS Router applies an array of routing algorithms to accurately route your board to completion.



PADS Router allows you to route even your most complex designs.

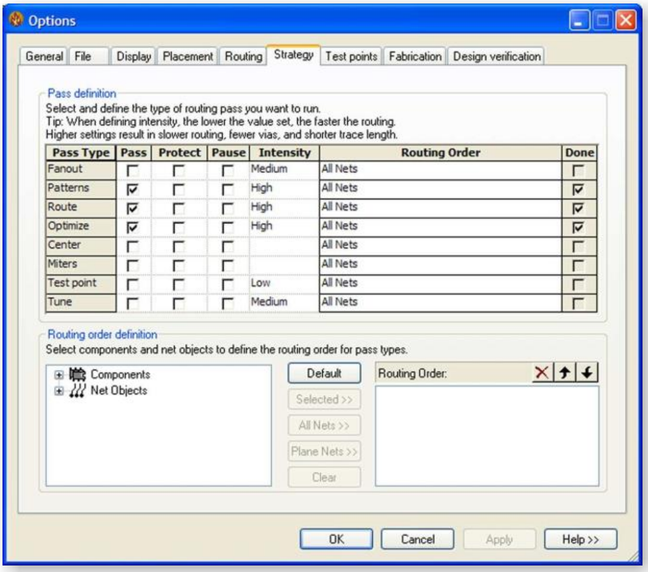
Partitioning Your Routing Tasks

PADS Router offers you the ability to partition your routing tasks into highly functional individual steps. Here's an example of how you might utilize this on areal-world design:

You begin by manually routing your critical high-speed nets such as clocks, andthen protecting them. Next you can run a fanout pass on your power nets to quickly connect power and ground to your planes. Once your critical and powernets have been routed, you can fan out each of your major components individually using the fanout options. After reviewing these operations, you canroute your data and address buses. Lastly, route the remaining nets to complete your design. You can also run Optimization and Centering passes as well as TestPoint passes to automatically insert test points into your design.

Routing Strategies

PADS Router Options allow you complete control over the routing process. You can select the order and intensity of each routing pass, thereby allowing yourself the flexibility to customize your strategy based upon your particular design needs. You can route individual components, nets, or your entire design.



PADS Router Strategies

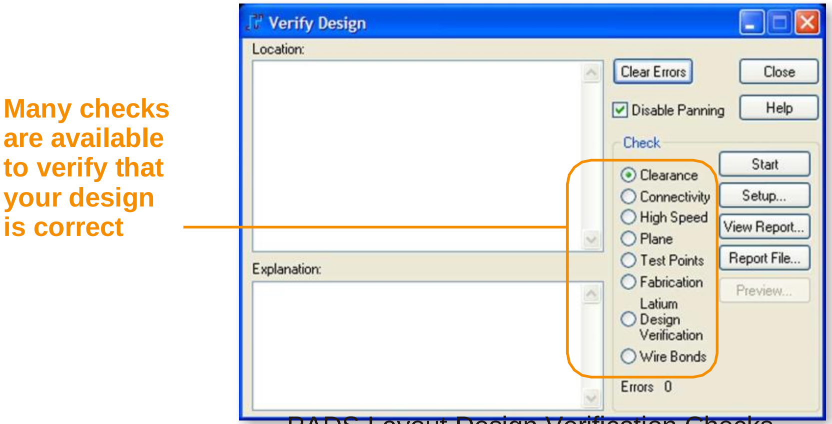
PADS Router utilizes a series of sophisticated route algorithms to maximize speed and accuracy when routing your designs. Once you familiarize yourself with the many routing pass options, you will quickly develop your own preferred strategy setups that are closely matched to your design needs. As you gain experience with PADS Router, you will find it to be a valuable design asset.

Design Verification

Design Verification

Before you send your design out for manufacture, you will want to check the accuracy of the final design. PADS offers a large selection of checks that you can initiate, including clearance checks, plane connectivity, trace widths, component body spacing, acid traps, soldermask slivers, and numerous other verification checks. PADS Layout and PADS Router each offer a wide range of checks.

The Verify Design options in PADS Layout allow you to perform clearance and connectivity checks on your design to assure both that proper clearances exist between design objects, and that all signals are connected. If you have plane layers in your design, you should run a plane check to make sure that all of your power and ground signals are properly connected to the plane layers.



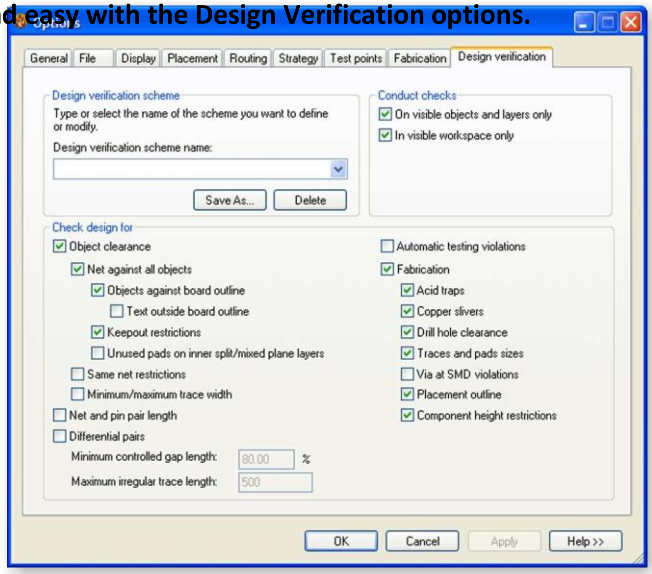
PADS Layout Design Verification Checks

Depending upon your design complexity, additional checks are available for high-speed signals, test points, and fabrication.

The Design Verification checks in the PADS products ensure that your designs will meet your most stringent manufacturing requirements. Checking your designs prior to sending them to the fabricator saves you time and keeps you on schedule.

Design Verification in PADS Router

If you have completed your design using PADS Router, you may want to take advantage of the advanced design verification checks available. PADS Router cancheck your differential pairs, min/max routes, and many other high-speed routingrules. Setup is quick and easy with the Design Verification options.



PADS Router Design Verification Options

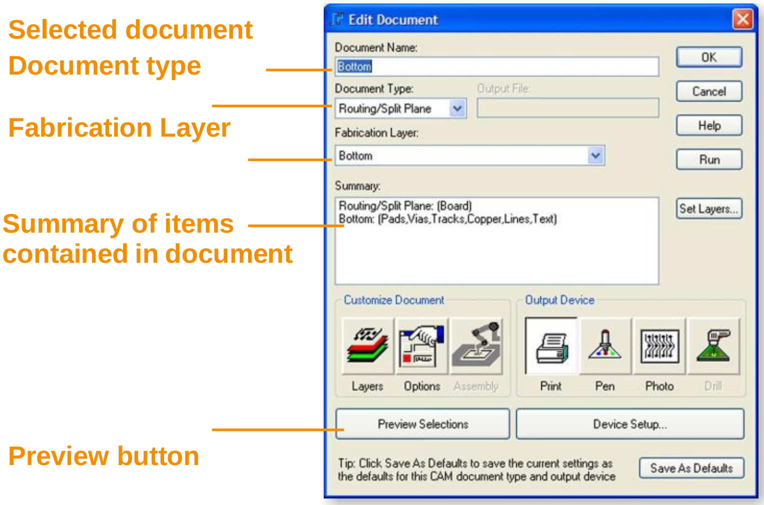
In PADS Router, errors can be viewed on-screen and in a spreadsheet view or a report where you can examine and determine your course of action for each item.Verifying your design before it goes to the fabricator helps to reduce manufacturing problems and keeps your project on schedule.

In this example, while routing certain differential pair signals, the designer chose to override some of the clearance rules to route escape traces near a connector. A Design Verification clearance check shows that there are multiple violations in the design. By selecting one of the errors in the spreadsheet view, the system will zoom in on the exact location of the error so that you can examine the error condition. This example shows that the traces are too close together. The error messages contain hyperlinks that will open a report containing specific information

CAM Documents

Generate the CAM Documents

After your design has been captured, routed, and checked, it's time to create the manufacturing outputs. Generating the Computer Automated Manufacturing (CAM) files is the last step in the process. PADS PCB Design Solutions support the creation of the full set of manufacturing outputs, including prints and plots, photoplots (Gerber files), drill files, pickand place data, soldermasks, paste masks, fabrication drawings, and assembly drawings.



Editing of a print document for the bottom layer of a design.

You can generate prints or plots of your design layers and choose the individual design objects that you want to include in each of the files. You can also scale and rotate the prints to fit your available printing devices.

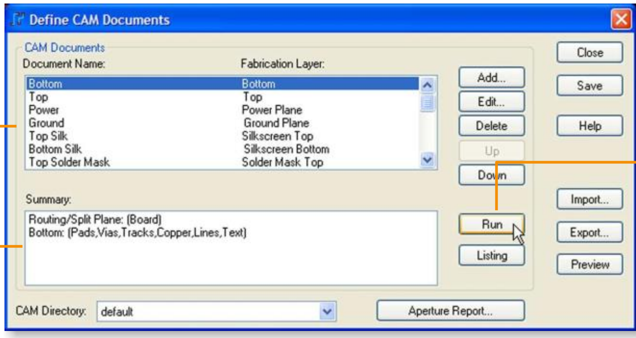
The advanced Computer Automated Manufacturing (CAM) file generation capabilities in PADS Layout allow you to quickly produce all of the files and documents needed to manufacture your design. CAM setups can be saved and reused on future design projects.

CAM Setup

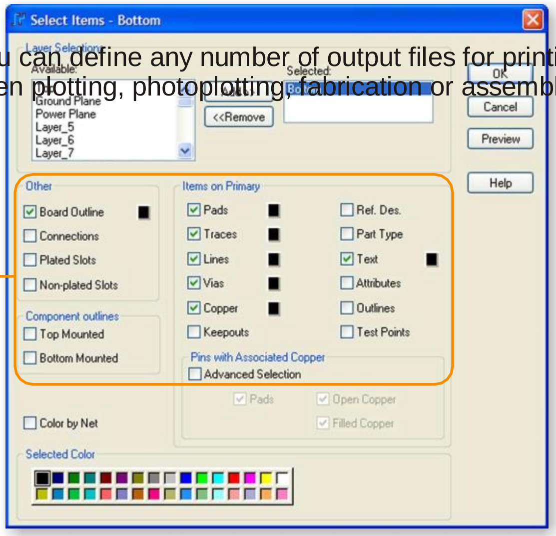
Shows the currently defined CAM documents

Shows the selections

for the current document



Use the Run button to generate the selected output files



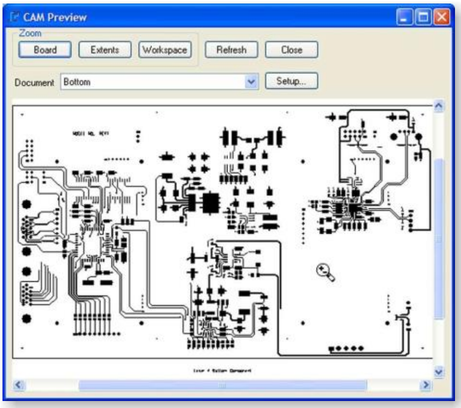
You can define any number of output files for printing, pen plotting, photoplotting, fabrication or assembly.

Specify each item that you want to appear in the selected output file

Specifying the CAM Options

Once your manufacturing preferences are defined, you can save them to apply to future designs. The Preview feature allows you to visually inspect your outputs before generating the final files and sending them out for manufacture. When you

CAM Preview



Use CAM Preview to inspect your outputs.

What’s Next?

Now that you've organized your information and taken a brief look at the design flow, you're ready to take things to the next step. We've only begun to illustrate the advanced capabilities available with PADS PCB Design Solutions; there is much more foryou to discover.

Remember, this document only introduces you to the PADS design process. It is not a comprehensive user guide. Your most complete source of detailed feature information is the Help in each application. In addition, use the other resources available from the Welcome screens, including QuickStarts, Product Tours, Tutorials, Concepts Guides, and links to web content such as technical documents and PADS Support.

Congratulations. You're well on your way to designing your first PADS project with confidenceand certainty.

References

QuickStarts

PADS Logic QuickStart
PADS Layout QuickStart
PADS Router Interactive Routing QuickStart
PADS Router Autorouting QuickStart

Concepts Guides

PADS Layout Concepts Guide
PADS Router Concepts Guide

PADS Logic Tutorials

Learning the User Interface
Creating Library Parts
Adding and Copying Parts
Adding Connections
Adding Buses
Editing Design Elements
Assigning Constraints
Creating Reports
Linking to PADS Layout
Updating a Schematic with Design Changes
Printing and Plotting Schematics
Managing Multiple Sheets

PADS Layout Tutorials

Learning the User Interface
Creating Library Parts
Preparing a Design
Importing Design Data
Importing Schematic Data
Assigning Constraints
Moving and Placing Components
Creating and Editing Traces Creating
Traces with Dynamic Route
Autorouting with PADS Router
Creating Split Planes
Creating Copper and Pour Areas
Adding Dimensioning
Checking for Design Rule Violations
Updating the Schematic
Creating Reports and CAM File

HINDU COLLEGE GUNTUR

CERTIFICATE

TO WHOMSOEVER IT MAY CONCERN

This is to certify that Mr. **………………………………………………….……….**
S/o **……………………**ennoblement no: **…………..** course **B.Sc** – semester IV student of **HINDU COLLEGE, UG COURSES**, Guntur, have completed his training on **PCB DESIGNING AND FABRICATION GUNTUR** from 25 – 02 -2021 to 04 – 07– 2022. During the tenure of his training, his conduct and contribution have been………………………

We wish him all the best.

PRINCIPAL

LIST OF PARTICULARS REGARDING PCB DESIGN AND FABRICATION

Name of the Add on course	:	PCB design and fabrication
Course duration	**:**	24 days
In charge	:	G.Siva Kumar
Trainee	**:**	Sai Saandeep gaaru
Group	:	Bsc Electronics
No.of students attended	**:**	20

Related pictures

RELATED PICTURES OF PCB DESIGIN

II-BSC MECS PCB ATTENDANCES

S.NO	NAME OF THE STUDENT	25/2	26/2	4/3	5/3	11/3	12/3	19/3	25/3	26/3	8/4	9/4	22/4	23/4	29/5	30/5	6/6	7/6	13/6	14/6	26/6	27/6	28/6	3/7	4/7
1	B.SAI SEKHAR	/	/	/	A	/	/	/	/	/	/	A	/	/	/	/	/	/	/	/	/	/	/	/	/
2	CH.SURYA ROHIL	/	/	/	/	/	/	/	/	A	/	A	/	/	/	/	/	/	/	/	/	/	/	/	/
3	CH.JYOTHIR MAY	A	/	/	/	/	/	/	/	A	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
4	K.JYOTHI	A	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
5	K.PAVANI	/	A	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
6	M.SIVA KOTESWAR A RAO	/	/	/	A	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
7	T.SUDARSH AN	/	/	/	/	/	/	/	/	A	/	A	/	/	/	/	/	/	/	/	/	/	/	/	/
8	D.MADHU BABU	/	/	/	/	A	/	/	/	/	/	A	/	/	/	/	/	/	/	/	/	/	/	/	/
9	P.NARENDR A	/	/	/	A	/	/	/	/	/	/	/	A	/	/	/	/	/	/	/	/	/	/	/	/
10	MALLIKARJ UNA CHARI	/	/	/	/	/	/	A	/	/	/	/	A	/	/	/	/	/	/	/	/	/	/	/	/
11	VENU BABU	A	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
12	SRINIVAS	A	/	/	/	/	/	/	/	A	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
13	SIVANJANE YULU	/	/	/	/	/	/	/	/	A	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/

14	ANIL KUMAR	/	/	/	/	**A**	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
15	UDAY KUMAR	/	/	/	/	**A**	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/
16	PAVAN	/	/	/	/	/	/	/	/	/	/	**A**	/	**A**	/	/	/	/	/	/	/	/	/	/	/
17	KALYAN	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	**A**	/	/	/	**A**	/	/	/	/
18	Y.RAJA	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	**A**	/	/	/	/	/	/	**A**
19	PRADEEP	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	/	**A**	/	/	/	**A**	/	/	/
20	NARAYANA REDDY	/	/	/	/	/	/	/	/	/	/	**A**	/	/	/	/	/	/	/	/	**A**	/	/	/	/